



Aalborg Universitet

AALBORG UNIVERSITY  
DENMARK

## Dilution in Transition Zone between Rising Plumes and Surface Plumes

Larsen, Torben

*Published in:*

Proceedings of the 3rd International Conference on Marine Waste Water Discharges and on Marine Environment / 1st International Exhibition on Materials Equipment and Services for Coastal WWTP Outfalls and Seelines : MWWD / IEMES 2004

*Publication date:*  
2004

*Document Version*  
Early version, also known as pre-print

[Link to publication from Aalborg University](#)

*Citation for published version (APA):*

Larsen, T. (2004). Dilution in Transition Zone between Rising Plumes and Surface Plumes. In *Proceedings of the 3rd International Conference on Marine Waste Water Discharges and on Marine Environment / 1st International Exhibition on Materials Equipment and Services for Coastal WWTP Outfalls and Seelines : MWWD / IEMES 2004: Catania, Italy, 27 September - 2 October 2004*

### General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal -

### Take down policy

If you believe that this document breaches copyright please contact us at [vbn@aub.aau.dk](mailto:vbn@aub.aau.dk) providing details, and we will remove access to the work immediately and investigate your claim.

## Dilution in transition zone between rising plumes and surface plumes

TORBEN LARSEN <sup>1</sup>

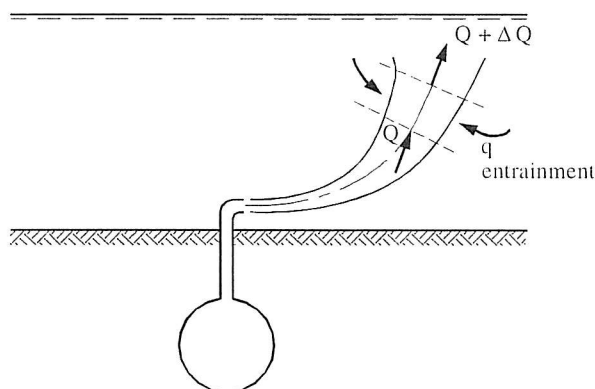
### Abstract

The papers presents some physical experiments with the dilution of sea outfall plumes with emphasize on the transition zone where the relative fast flowing vertical plume turns to a horizontal surface plume following the slow sea surface currents. The experiments show that a considerable dilution takes place in this zone depending on the ratio between the velocities of the inflow and the outflow of the zone. The option of taking this increased dilution into consideration in respect to initial dilution requirements is discussed.

### The zones of sea outfall dilution

When sewage is discharged from a sea outfall diffuser in shallow water the plume rises towards the water surface in the so-called initial dilution zone because of density difference between the sewage and the seawater. This difference will on most coasts be in the order of  $15 - 25 \text{ kg/m}^3$  equal to 1.5 - 2.5 percent of the total density.

#### *THE RISING PLUME*



*Figure 1. The rising zone*

---

<sup>1</sup>Torben Larsen, professor, dr.techn., Aalborg University, Department of Civil Engineering, Sohngaardsholmsvej 57, DK-9000 Aalborg, Denmark, telephone +45 96358481, e-mail [torben.larsen@civil.aau.dk](mailto:torben.larsen@civil.aau.dk) . web [www.aau.dk](http://www.aau.dk)

When the sewage leaves the diffuser port it contains buoyancy as well as momentum. Due to the buoyancy the plume tends to rise towards the surface. But because of the turbulent dilution and entrainment of the ambient water the buoyancy is gradually reduced.

The dilution in the rising zone has been studied for a half a century and the literature contains famous authors in the field of environmental hydraulics as for example Morton, Taylor and Turner [1], Abraham[2], Cederwall [3], Fan and Brookes [4], Jirka and Harleman [5].

#### THE TRANSITION ZONE

When the plume hits the surface it forms a so-called “bowl” or transition zone where the plume transforms to a co-flowing surface plume (figure 2).

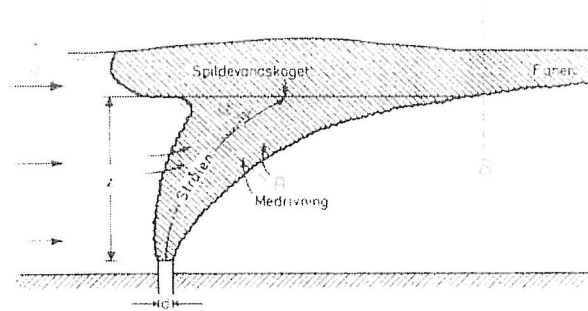


Figure 2 The transition zone

Depending on the current in the ambient sea an upstream wedge can be formed [6]. The criterion for the existence of this wedge is in principle that the surface plume can be formed with a densimetric Froude number  $F_d$  equal to one:

$$F_d = \frac{u_a}{\sqrt{\frac{\Delta\rho}{\rho} gh}} \approx 1 \quad (1)$$

where  $u_a$  is the ambient current velocity,  $\frac{\Delta\rho}{\rho}$  is the relative density difference between plume and ambient sea,  $g$  is the gravity constant and  $h$  is the plume height.

In the engineering design of outfall diffusers the dilution in the transition zone where the plume turns from the rising phase to establish a buoyant co-flowing surface plume is most often neglected. The aim of this note is to present experimental results on the dilution in this transition zone.

A free rising plume can hydraulically be understood as a super critical flow with a densimetric Froude number equal to infinity, whereas the co-flowing rising plume is a sub critical flow. Accordingly the transition zone could be understood as an internal hydraulic jump where a significant entrainment and dilution takes place.

### THE SURFACE PLUME

The surface plume will in practise always follow the current in ambient sea water. Near the “bowl” a certain density difference will exist but gradually this difference will vanish when the plume is moved away, and after some hundreds of meters the transport and dilution can be understood as a passive transport totally controlled by the currents and the turbulence in the sea water.

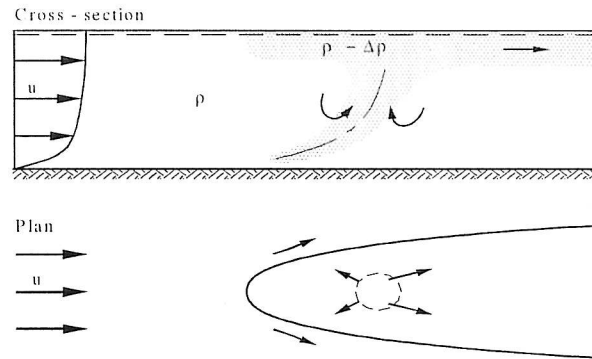


Figure 3 The surface plume

The author has presented a rather simple so-called integral model was presented which can describe surface plume in both the first buoyant phase and the later turbulent phase [7].

### Dilution in the transition zone

In order to study the dilution in the transition zone a number of physical and a few numerical experiments has been carried out.

#### EXPERIMENTAL SET UP

The experiments took place in a re-circulating laboratory flume with a length of 15 m, width of 1.5 m and a depth of 0.40 m. The density difference in the plume was adjusted by adding sodium chloride to the ambient water. The plume was formed by a vertical discharge of fresh water from a circular hole at the bottom of the plume. The discharged fresh water was given a higher temperature than the saline water and the dilution was found from measurement of temperatures in the plumes and the ambient water by a thermo couple equipment. Details on the experiments shall not be given here but the range of the variables was as follows:

Water depth in flume	0.15 – 0.20	m
Ambient flow velocity	0 – 0.1	m/s
Nozzle diameter	12 and 17	mm
Nozzle velocity	0.05 – 0.3	m/s
Density difference	15 – 35	kg/m <sup>3</sup>

## EXPERIMENTAL RESULTS

Only results for the case of a stable (upstream) bowl are given. In figure 2 the inverse dilution  $M = l/S$  is shown as function of the ratio  $V = U_c/U_a$ , the ratio between the inflow velocity and the outflow velocity to the transition zone.  $U_c$  is the average velocity (approx. half the centreline velocity) where the rising plume enters the bowl zone.  $U_a$  is equal to the ambient current velocity.

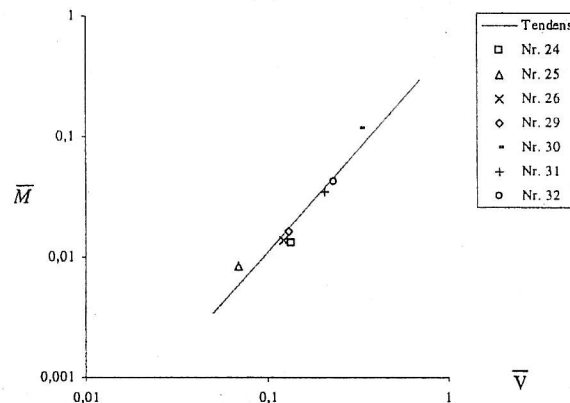


Figure 2. Centerline dilution  $S$  as function of relative rising plume velocity

It is obvious that the dilution increases with increasing difference in velocity corresponding to the way the head loss increases in an ordinary free surface hydraulic jump.

In principle the graph cannot be extrapolated down to low values of the relative rising plume velocity. In this case of stagnant ambient flow the case turns to be an unsteady circular surface plume with an increasing height. Unfortunately this makes the results obtained in this study less relevant in practice, when the initial dilution requirements are related to stagnant waters.

### 3-dimensional numerical modelling in progress

Formulas and diagrams developed from physical model experiments have been the normal basis for the practical design of sea outfalls. Also a number of so-called integral models have been developed [8]. But 3-dimensional numerical flow modelling is an obvious possibility for the design of solutions which are not covered by the traditional methods. In this section the liability of numerical models in respect to the dilution in the transition zone is discussed.

The numerical modelling is performed by the MIKE 3 software packet (distributed by DHI, Water and Environment), which is a general 3-dimensional hydrodynamic flow model solving the Navier Stokes equations allowing various turbulence descriptions for calculating the shear stresses. In the actual computations the standard k- $\epsilon$  turbulence model was applied taking into account the influence of the density differences.

A considerable number of turbulence models exist. These models are grounded on various simplified theoretical assumptions for the turbulence. But it is important to understand that they all basically are building on a rather empirical adaptation to a number of characteristic physical experiments. It is only from practical experience one can judge whether a certain

model is adequate for a specific task.

It is accepted that the k- $\epsilon$  turbulence model can describe the turbulent mixing adequate under stable stratification. In this connection has the author and co-authors shown that the k- $\epsilon$  turbulence model seems adequate for computation of the dilution of both a surface plume and a dense bottom plume [8]. For the unstable situation as the free rising turbulent plume, it has been shown that this model can describe the dilution with a reasonable accuracy [9].

In the actual situation where the rising plume enters the surface and continues as a sub critical plume it cannot a priori be expected to be coped by a turbulence model. The literature unfortunately does not give reference to comparison of measurements and computations of the dilution in submerged hydraulic jumps in stratified flows. A study in progress is trying to validate the k- $\epsilon$  turbulence model in respect to the dilution in “hydraulic jump-like” flows.

Anyhow, it should be mentioned that it is the authors general opinion that the 3-dimensional turbulence modelling for inhomogeneous fluids still is less accurate than physical modelling with in scale models. It is often seen that that the numerical models underestimates the dilution with 20 – 30 %.

### **Conclusion and some words on implications for initial dilution requirements**

The results indicate that a significant dilution takes place in the transition zone.

The author has recently argued [10] that the need for initial dilution in respect to environmental problems seems reduced because of the high demands for sewage treatment for example in Europe because the treatment nowadays reduces the concentration of the pathogens (bacteria and vira) with a factor up 1000 times compared to the old days when the initial dilution criteria was settled. A reduction in the initial dilution demands will lead to more simple and reliable design of the diffusers [7] both in respects to proper functioning of the diffuser and to the environmental performance. The findings in this study support this statement further.

### **Acknowledgements**

Part of the experimental works described in this paper was carried out in the final semester in a MS study by Mr. Stefan Sommer. Stefan is acknowledged for his excellent work in the laboratory.



AALBORG UNIVERSITY

Department of Civil Engineering  
Sohngaardsholmsvej 57, DK-9000 Aalborg, Denmark  
Web: [www.aau.dk](http://www.aau.dk)

Department of Civil Engineering at Aalborg University covers the following sections:

- Marine and Environmental hydraulics
- Soil mechanics
- Structural mechanics and dynamics
- Physical geography

### References

- [1] Morton, B.R., Taylor, G.I. and Turner J.S. (1956). Turbulent Gravitational Convection from Maintained and Instantaneous Sources. Proc. Royal Society of London. A234.
- [2] Abraham, G. (1963). Jet diffusion in stagnant ambient fluid. Delft Hydraulics Laboratory. Publ. no. 29.
- [3] Cederwall, K. (1968). Hydraulics of marine waste water disposal. Report no. 42, Hydraulics division. Chalmers Institute of Technology, Gothenburg.
- [4] Fan, L.N. and Brooks, N.H. (1969). Numerical solutions of turbulent buoyant problems. W.M. Keck Laboratory of Hydraulics Report No.KH-R-18, California Institute of Technology,
- [5] Jirka, G. and Harleman, D.R.F. (1973). The mechanics of submerged multiport diffusers for buoyant discharges in shallow waters. R.M. Parsons Laboratory for Water resources and Hydrodynamics Report. No. 169, MIT.
- [6] Chen, H.B. and Larsen, T. (1995). Surface intermediate zone of submerged turbulent buoyant jet in current. *Proc. XXVIth IAHR Congress - HYDRA 2000*, London., September 1995.
- [7] Larsen, T. (1994). Diffuser design for marine outfalls in areas with strong currents, high waves and sediment transport. *Wat. Sci. Tech.* Vol. 32., No. 2, pp 249-255.
- [8] Petersen, O. and Larsen, T. (1990). Laboratory study of dispersion of buoyant surface plumes. Proc. Int. Conf. Physical Modelling of Transport and Dispersion, MIT, August 7 - 10, 1990
- [9] Chen, H. B., Larsen, T. and Petersen, O. (1991). Turbulent buoyant jets in flowing ambients, *Environmental Hydraulics*, Lee & Cheung (eds) 1991.
- [10] Larsen, T. (2000). Necessity of Initial Dilution for Sea Outfall Diffusers in Respect to the European Directive on Municipal Discharges. Proc. International Conference on Marine Waste Water Discharges 2000, Genoa, November 2000.